7.0 OUTPUT

Output is found in the code-generated files (output file, write file, history plot file, solute plot file, contour plot file, contour plot file for dual or dpdp**,** stiffness matrix data, input check file, and AVS output files) described in Section 5.0.

Macro commands (input options) dealing with output control are **cont** (page 24), **ctrl** (page 27), **nod2** (page 42), **node** (page 42), and **time** (page 60). The command **cont** is used to specify output format and time intervals for contour data output (*fehmn.con*, *fehmn.dp*); **ctrl** is used to specify if element coefficients calculated in the code should be saved (*fehmn.stor*); **node** and **nod2** are used to provide nodal or coordinate positions for which general information and time history data will be output (*fehmn.out*, *fehmn.his*, *fehmn.trc*, and terminal output); and **time** provides input on the time printout interval for nodal information (*fehmn.out* and terminal output).

The code itself provides no graphical capabilities. A software environment, called the Browser, provides preprocessing, processing, and postprocessing capabilities and a help-index for FEHM. A complete description of how to use the Browser is given in Section 8.5.1. The Browser Pre-Processor assists the user in setting up an input file by displaying the appropriate pages from this User's Manual, doing syntax checks, analyzing the input file, and providing direct access to auxiliary programs for structured and unstructured grid generation.

The Browser Processor allows the user to run FEHM, restart it, and provides assistance in setting up a control file.

The Browser Post-Processor provides FEHM time history plots, contour plots, and access to AVS (an AVS license is required for its use).

Time history plots of the energy source, source strength, temperature, pressure, capillary pressure, and saturation are made from the *filen.his* FEHM output files. Data from the *filen.trc* files are used to make time history tracer plots of the 10 species concentrations. Capabilities for zooming, scaling, using a log scale, and printing the data are provided.

Contour plots using 2-D quad grids and 3-D hex grids for material properties, temperature, saturation, pressure, velocities, and solute concentrations can be made from the AVS FEHM output files. The plots can be rotated, zoomed, scaled, translated, and printed. Axis values and the color bar can be customized.

AVS provides tools for visualization and analysis of volumetric scalar and vector data. AVS FEHM output files are available for the following node data: material properties, liquid- and vapor-phase values, velocity and pressure values, temperature, saturation, concentration dual, and dpdp. The AVS output files from FEHM are written in either an ASCII or binary format that can be imported into AVS UCD graphics routines for viewing.

Two additional graphical postprocessing routines are available for use with output files *filen.his*, *filen.trc*, and *filen.con*, as discussed in Section IIIC of Zyvoloski et al. (1991).

Additional information on the data found in the output files is given below.

7.1 Output File (*filen.out*)

Information contained in the general output file is mostly self explanatory. The file starts with the code version, date, and time followed by the user input problem title. A summary of the I/O files used, macro control statements read, and array storage follow. Variable information for user-specified nodes at user-selected time intervals is written. The file ends with a summary of simulation time, number of time steps in the problem, the number of iterations taken, and total cpu time.

7.2 Write File (*filen.fin*)

The write file contains the final values of pressure, temperature, saturation, and simulation time for the run. The final version of the file is generally written at the end of the simulation. This information is also written if the minimum user-supplied time step has been realized and execution is terminated. If the write file has not been specified at startup, the code will use *fehmn.fin*. The primary use of the write file is as a restart file. The write file contains the following:

Code version number, date, time

Problem title

Simulation time (days)

Gas flag (ngas, h20, air)

Tracer flag (trac, ptrk, ntra)

Stress flag (strs, nstr)

Dpdp flag (dpdp, ndpd)

Dual flag (dual, ndua)

If ngas flag is set, followed by:

Final temperature (°C) at each node

Final saturation (dimensionless) at each node

Final pressure (MPa) at each node

Final capillary pressure (MPa) at each node

Or if neither the air or ngas (h20) flag are set, followed by:

Final temperature (°C) at each node

Final saturation (dimensionless) at each node

Final pressure (MPa) at each node

Or if air flag is set, followed by:

Final saturation (dimensionless) at each node

Final pressure (MPa) at each node

If trac flag is set, followed by:

Number of species

Species concentration (dimensionless) for each node for each species

Or if ptrk flag is set, followed by:

Number of particles, final random number seed

Final node position for each particle (if the value is negative, the particle left the model domain at a fluid sink at that node)

Fractional time remaining at current node for each particle

Multiplier to the plug flow residence time for each particle at the current node position, accounting for dispersion, sorption, and matrix diffusion effects

Age for each particle, i.e., the time since the particle entered the system. However, if the particle has left the system, this value is the time the particle left.

If the random number seed in the file is negative, the arrays for the fractional time remaining and the multiplier to the plug flow time have been omitted using the PRNT_RST = -1 option (see PRNT_RST description in the PTRK macro). A restart simulation using this input file will only approximate the behavior of particles because each particle will be assumed to have just entered the node. It is preferable to restart a particle-tracking simulation using a file that contains the full restart information.

If strs (not implemented in this version)

If dpdp or dual flag was set:

The above information including dual-porosity/dual-permeability nodes

7.3 History Plot File (*filen.his*)

The history plot file contains the following:

Code version number, date, time

Problem title

Tracer flag ('trac' or blank)

Stress flag ('strs' or blank)

Number of nodes for which data are output

Node number and x-, y-, and z-coordinate (m) of each node for which data are output

'headings'

'node flow enthalpy (Mj/kg) flow (kg/s) temperature (°C) total pressure (Mpa)'

'capillary pressure (MPa) saturation (kg/kg)'

And for each time step:

Time (days) followed by:

Node number, energy source (MJ/s), source strength (kg/s), temperature (°C), pressure (MPa), capillary pressure (MPa), saturation (dimensionless) for each specified output node

7.4 Solute Plot File (*filen.trc*)

Solute data are output for the same nodes used for the history plot file. The solute plot file contains:

Code version number, date, time

Problem title

Number of nodes for which data are output

Node number and x-, y-, and z-coordinate (m) of each node for which data are output

Number of different species for tracer solution

And for each time step:

Time (days), species number followed by

Species concentration (dimensionless) for each specified output node.

When particle tracking is used, the concentration can be output in several different forms (number of particles, number per fluid mass, or number per total volume). The choice of which form to use is input in the **ptrk** macro.

7.5 Contour Plot File (*filen.con*)

The contour plot file contains:

Code version number, date, time

Problem title

Tracer ('trac') solution or blank

Stress ('strs') solution or blank

Number of nodes for which data are output

X-, y-, and z-coordinate (m) of each node for which data is output

Number of nodes per element, total number of elements

Nodal connectivity information for each node of each element

X-, y-, and z-permeability (m^2) for each node

X-, y-, and z-thermal conductivity $\left(\frac{W}{m \cdot K}\right)$ for each node

Porosity, rock specific heat $\left(\frac{MJ}{kg \cdot K}\right)$, capillary pressure (MPa) for each node

Number of degrees of freedom per node for the current problem, direction of gravity in problem, value of gravity

If tracer solution is present:

Number of species

And for each specified time:

Time (days), injection phase (≥ 0 liquid, < 0 vapor) followed by

If injection phase is liquid:

Liquid transmissibility/density, liquid density (kg/m^3), pressure - capillary pressure (MPa), temperature (°C)

And if tracer solution is present:

Species concentration of liquid phase

Or if injection phase is vapor:

Vapor transmissibility/density, vapor density (kg/m^3), pressure (MPa), temperature (°C)

And if tracer solution is present:

Species concentration of vapor phase.

7.6 Contour Plot File for dual or dpdp (*filen.dp*)

The contour plot file for dual or dpdp contains the same information as the regular contour plot file only the parameter values are for the dual-porosity/dual-permeability nodes.

7.7 Stiffness Matrix Data (*filen.stor*)

The stiffness matrix data file is used to store the finite-element coefficients for each node. The file eliminates the need for the code to recompute the coefficients for subsequent runs. It contains the following:

Code version number, date, time

Problem title

Number of storage locations needed to store geometric input types, number of nodes, size of connectivity array

Volume associated with each node

Nodal connectivity information for each connection

Position of geometric coefficient for each connection

Diagonal position in connectivity array for each node

Finite-element geometric coefficient for each for each storage location

If stress solution is enabled:

Finite-element geometric coefficient for each for each storage location for the stress module.

7.8 Input Check File (*filen.chk*)

This file contains a summary of input information that may be of use in debugging or memory management of the code. The positions of maximum and minimum values of input parameters and derived quantities are given. Also provided is an analysis of array storage requirements.

7.9 Error Output File (*fehmn.err*)

This file contains the code version number, date, and time followed by any error or warning messages issued by the code during a run.

7.10 AVS Log Output File (*filen.10001_avs_log*)

The AVS log output file contains:

Code version number, date

AVS log identifier

Problem title

And for each specified time:

AVS output file prefix, call number, and time (days)

7.11 AVS Header Output Files (*filen.number_type_head*)

The data types are given in Section 7.13.1.

7.11.1 ASCII header

The AVS ASCII (formatted) header files contain:

20 lines of text with information about the FEHM AVS output files. The text is followed by a one-line AVS UCD file header containing:

number of nodes

number of cells

number of data components for the nodes

number of data components for the cells (currently 0)

number of data components for the model (currently 0)

7.11.2 Binary header

The AVS binary (unformatted) header files consist of 21 bytes with the following values:

number '7' indicating binary file (1 byte unsigned char)

number of nodes (4 byte int)

number of cells (4 byte int)

number of node data (4 byte int)

number of cell data (4 byte int) (currently 0)

number of model data (4 byte int) (currently 0)

7.12 AVS Geometry Output File (*filen.10001_geo*)

7.12.1 ASCII geometry output file

The ASCII (formatted) geometry file contains the following:

Node id and x-, y-, z-coordinate for each node

Cell id, material id, cell type, and the list of cell vertices

7.12.2 Binary geometry output file

The binary (unformatted) geometry file contains the following:

Number of nlist nodes (4 byte int)

Cell id, material id, number of nodes, cell type (16 * num_cells) (ints)

Cell vertice list (4 * num_nlist_nodes) (ints)

X-coordinates for nodes (num_nodes * 4) (floats)

Y-coordinates for nodes (num_nodes * 4) (floats)

Z-coordinates for nodes (num_nodes * 4) (floats)

7.13 AVS Data Output Files (*filen.number_type_node*)

7.13.1 ASCII node data output files

All the ASCII (formatted) node data files contain the following:

Number of data components and size of each component

A label/unit string for each data component

And for each node:

The associated node data (described by data type below)

7.13.1.1 Material properties *(_mat and _mat_dual)*

These data will consist of 11 fields. The order of the fields are:

Permeability in x-, y-, and z-direction (m^2)

Thermal conductivity in x-, y-, and z-direction $\left(\frac{W}{m \cdot K}\right)$

Porosity

Rock specific heat $\left(\frac{MJ}{kg \cdot K}\right)$

Capillary pressure (MPa)

Relative permeability model

Capillary pressure model

The dual or dpdp values for each of these fields will be written to a file with “mat_dual_node” appended to the file name.

7.13.1.2 Scalar parameters *(_sca and sca_dual)*

These data files will contain scalar data including:

Saturation

Temperature (°C)

Liquid pressure(MPa)

Vapor pressure(MPa)

If dual values are calculated, they can be written to the sca_dual output file.

7.13.1.3 Vector parameters *(_vec and _vec_dual)*

These data files contain the vector values for:

Liquid and vapor velocities (m/s)

If idualp is defined, dual-porosity values for the vapor phase will be written to the _vec_dual file. If idpdp is defined, double-porosity/double-permeability values for the liquid phase will be written.

7.13.1.4 Solute concentrations *(_con and _con_dual)*

Up to 20 fields per node can be written for solute concentrations. The number written is determined by the number of species. The dual counterparts to each will be written to the _con_dual file.

7.13.2 Binary node data output file

All the binary (unformatted) AVS data files contain the following:

Node data labels (1024 byte string)

Node data units (1024 byte string)

Number of node components (4 byte int)

Node component list (num_node_data * 4) (ints)

Minimums for node data (num_node_data * 4) (floats)

Maximums for node data (num_node_data * 4) (floats)

Data blocks with values for each node (num_nodes * num_node_data * 4) (floats)

The data types are described above in Section 7.13.1.

8.0 SYSTEM INTERFACE

8.1 System-Dependent Features

In addition to standard intrinsic math routines, only two system routines are required by the FEHM code. The code uses a system call to get the date (subroutine dated) and a system routine to get the CPU clock time (subroutine tyming).

8.2 Compiler Requirements

FEHM is written in Fortran 77 and C. FEHM has been successfully compiled and run on SUN, HP, IBM RISC, SGI, and Cray computers.

8.3 Hardware Requirements

No special hardware features or environments are required by the software.

8.4 Control Sequences or Command Files

None.

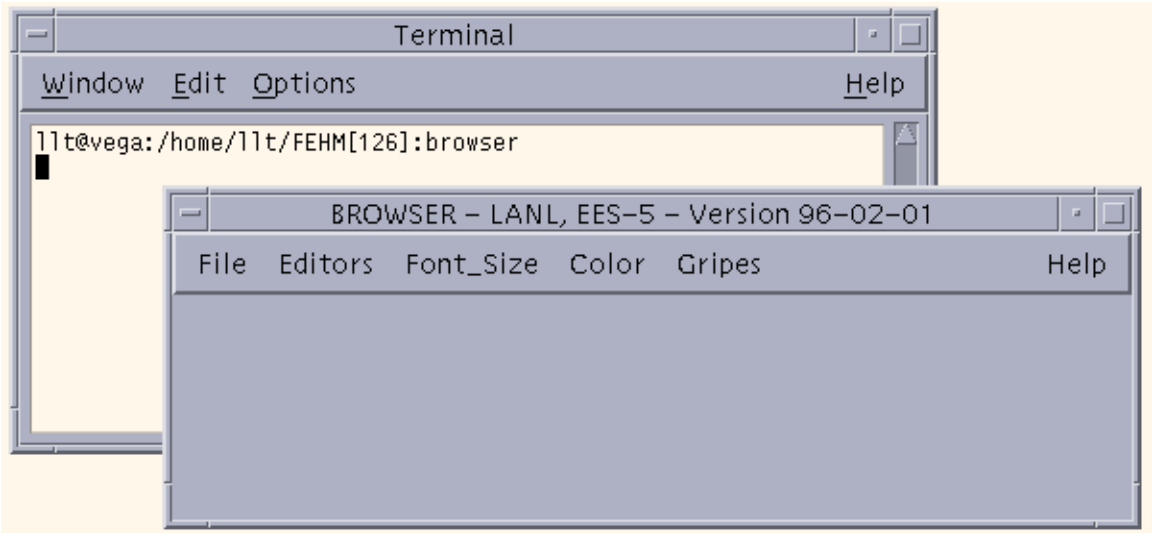
8.5 Software Environment

FEHM can be run under a software environment called the Browser. The Browser is a column-oriented graphical user interface that provides preprocessing, processing, and postprocessing capabilities and a help index for FEHM.

8.5.1 Browser

The Browser Pre-Processor assists the user in setting up an input file to be run with the Processor and provides direct access to auxiliary programs for grid generation. The Processor allows the user to run FEHM and restart it. The Post-Processor allows the user to view ASCII output files, make time history plots, make contour plots, and display AVS output files generated by FEHM (user must have an AVS license). There is also a help index that includes documentation for FEHM and auxiliary programs. To bring up the main Control Window for the Browser under X-windows, type:

browser

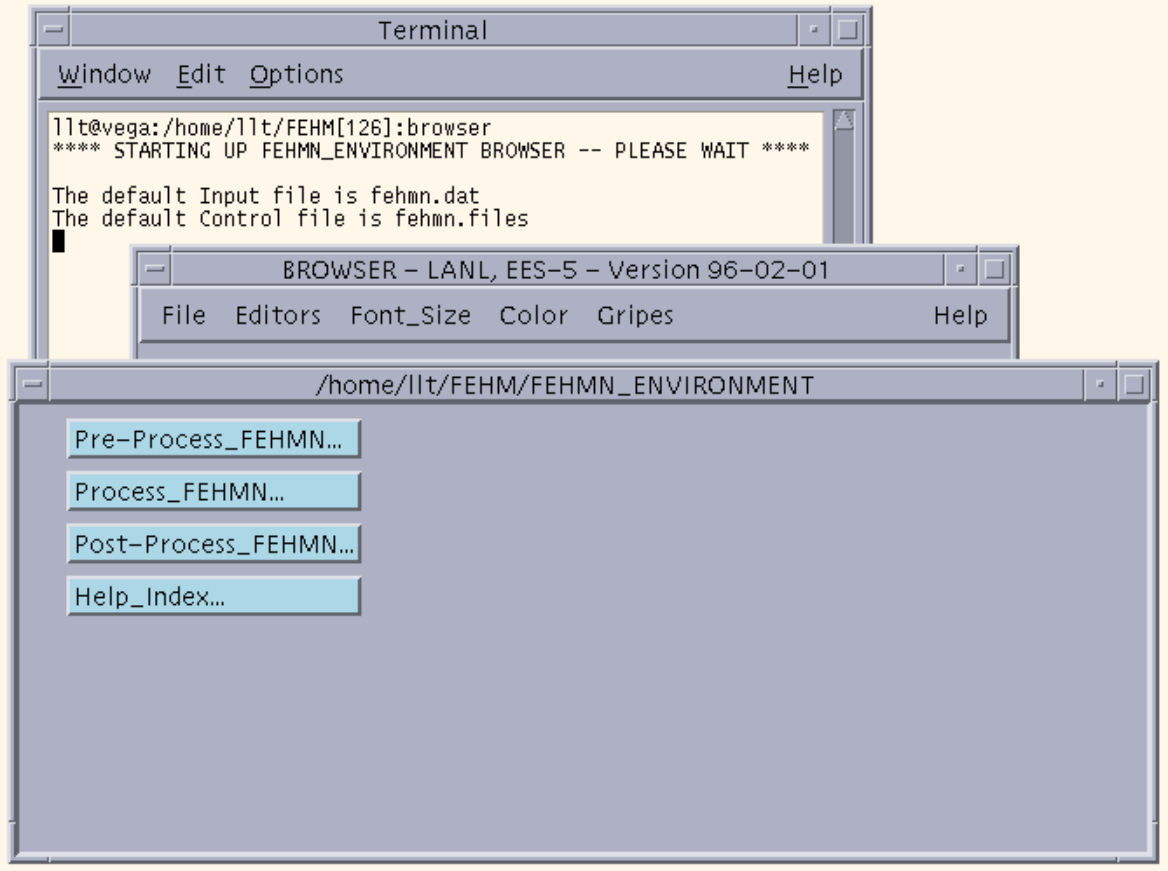


• Control Window

The *Editors* button along the top allows the user to switch editors. The default editor is*vi* . The editors *emacs* and *fred* are menu options for the Suns. *Jot* is a menu option for the SGIs. The *Font_Size* button allows the user to change the font size. *A* medium-size font is the default. The *Color* button allows the user to use the Browser on a black-&-white terminal. The *Gripes* button allows the user to report bugs and view a list of current bugs.

The *Help* button has a *Getting_Started* menu option that gives more details on using the above buttons. The *Help* button also allows the user to turn the man pages on or off. The default is to have the man pages on.

To bring up the Browser window, select *Open* in the Control Window under *File*, highlight the file *FEHMN_ENVIRONMENT*, and click on the *OK* button. The following Browser Window will appear.



• Browser Window

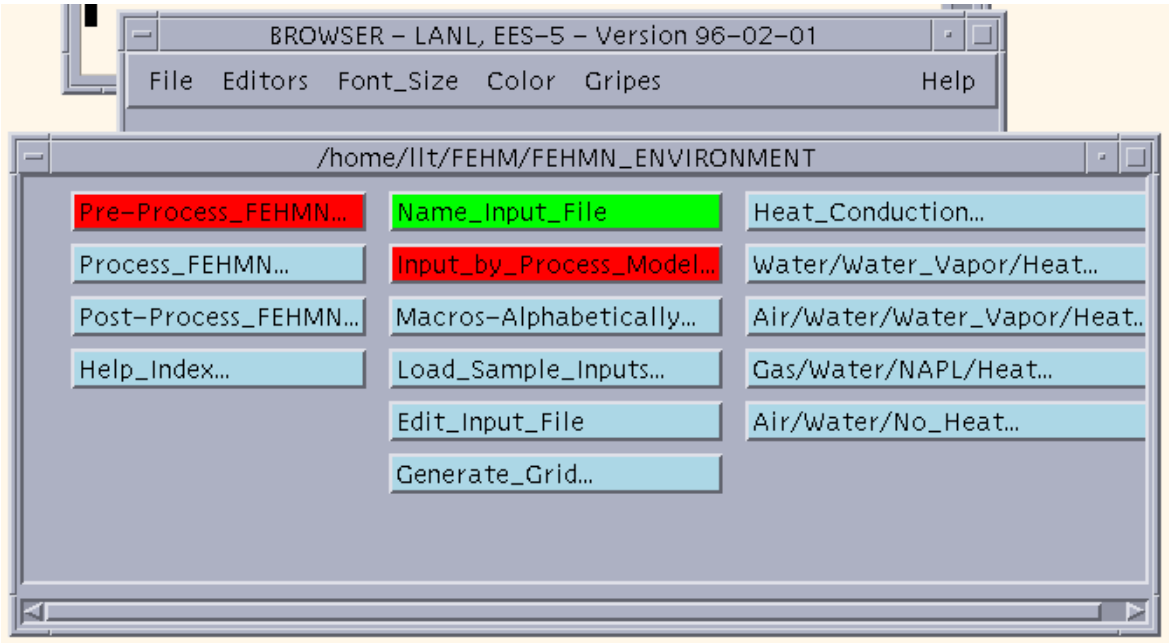
Use the Browser by clicking ONCE with the LEFT mouse button on the desired button. If the button name is suffixed with "...", another column of buttons will appear when selected. Otherwise, a function will be executed. Red buttons are the last buttons selected. Green buttons were pushed previously. Grey buttons have not been accessed.

The scroll bar can be used to reach buttons outside the displayed window, or the window can be resized by dragging the corner of the window.

When finished, quit the Browser from the Control Window, under *File*, by selecting *Exit*.

8.5.1.1 Pre-process FEHM

The *Pre-Process_FEHMN* button provides guidance, syntax checking, analysis, and help in creating an input file. Several auxiliary grid-generating programs are provided. Click on *Pre-Process_FEHMN* to get the column with these options.



- Create Input File

  The default input file name is *fehmn.dat*. To change this, the user should click on the button *Name_Input_File*. This file name must be assigned before working on the file. There are four ways the user can work on this file. A user can input by process model type, from a list of macros, by loading a sample input file, or by editing the input file directly. A combination of these ways can be used.

  The *Input_by_Process_Model* button allows the user to select the type of problem from five process models and guides the user through the required and optional macros for that process type. In the example above, the *Heat_Conduction* process model was selected. The user clicks on the *Required_*_Inputs* button to get a list of the macros required for a specific process type. When a button is clicked for a specific macro, the appropriate pages from the FEHM User's Manual are displayed and an editing window is opened for the user to input the macro. The input of macros are appended to the input file. A macro may be written to this file more than once. Displaying the FEHM User's Manual pages can be turned off under *Help* on the main Control Window. The editor can be switched by clicking on the *Editors* button in the main Control Window. As input is entered, a syntax check is made for the correct number and type of arguments. Corrections are made, if possible, and displayed. The *Analyze_Input* button can be used

to verify that all the required macros are present and to list the appropriate optional macros that have not been included. There is an option, the *Use_Sample_File_** button, for using an example input file for each process type. The example input file is loaded into this file. The *Edit_Input_File* button gives the user direct access to this input file at any time.

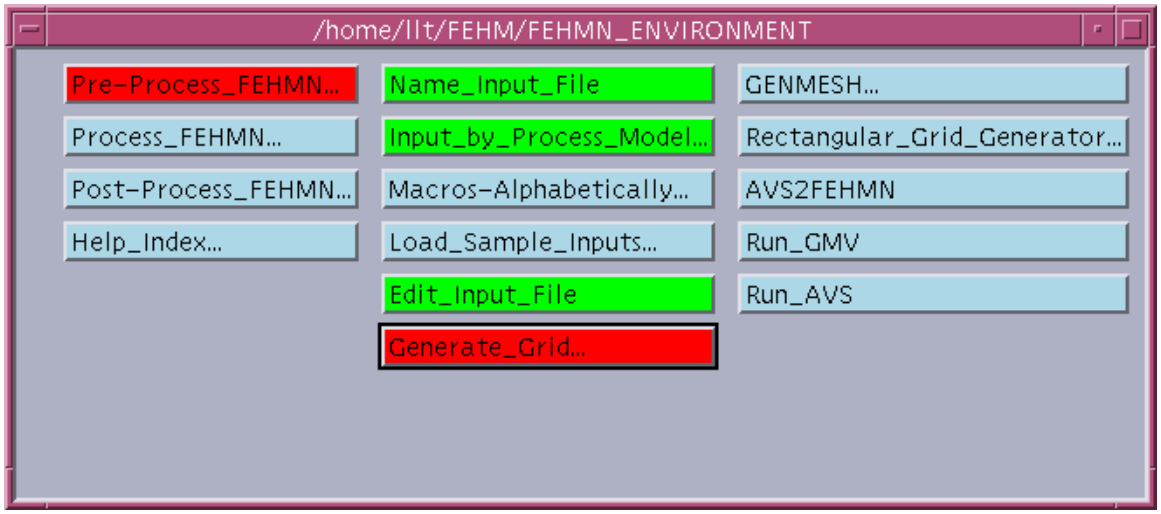
The *Macros-Alphabetically* button provides an alphabetical listing of all the macros. This mode will automatically bring up the FEHM User's Manual pages and an editing window. Macros entered are appended to the input file, and a syntax check is made on the input. Direct access to the input file is also available through the *Edit_Input_File* button. There is no input guidance or analysis of required and optional macros.

The *Load_Sample_Inputs* button replaces or creates a new input file. Direct access to the input file is available by going through the *Edit_Input_File* button. Access to the User's Manual is available through the *Help_Index* in the first column.

The *Edit_Input_File* button allows the user to directly edit the input file. The editor can be changed with the *Editors* button in the Control Window. The User's Manual is available by going through the *Help_Index* in the first column. There is no syntax checking, input guidance, or analysis on the input.

- Generate Grid

  Direct access is given to several auxiliary grid-generating programs. A brief description is given below. Additional help is available under the *Help_Index* button.



GENMESH is an automatic mesh-generation code that directly generates the finite-element mesh input required by FEHM. Help pages and an editing window are used to create the input file. Examples of input files for 2-D and 3-D grids are provided. A User's Manual is available in the Browser.

Rectangular_Grid_Generator program creates a structured grid in the format for an FEHM input file. If the file *input.grid* is

constructed, it can be run with an input file. Otherwise, an interactive mode is available that prompts for the necessary inputs.

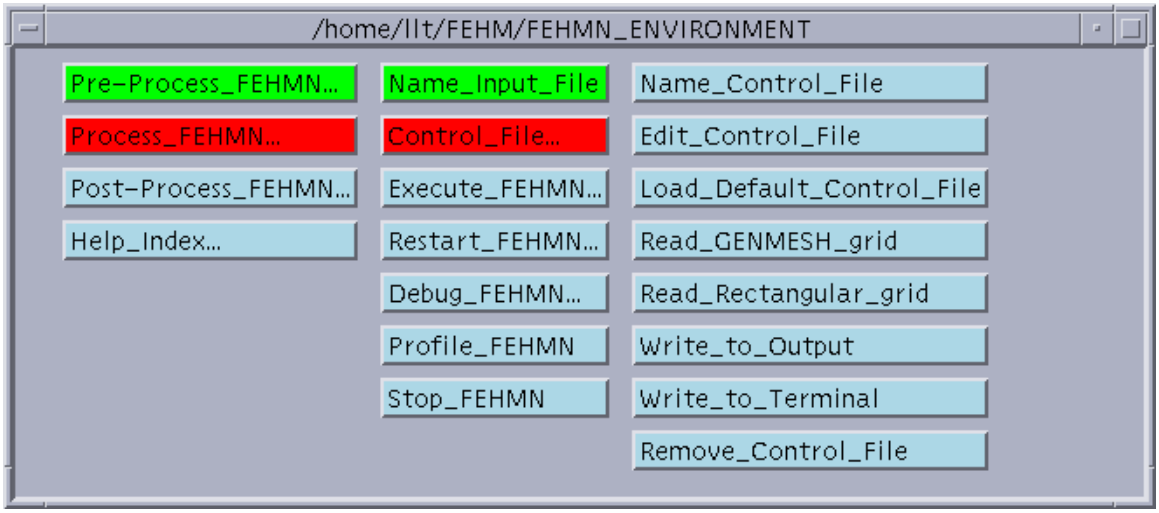
AVS2FEHMN takes an AVS file and converts it to the format for an FEHM input file.

Run_GMV is a General Mesh Viewer, a visualization tool that provides 3-D interactive graphics for data from any 3-D mesh.

Run_AVS executes the commercial program AVS. (An AVS license is required.)

8.5.1.2 Process FEHM

The *Process_FEHMN* button provides assistance in creating a control file and in executing, restarting, and stopping FEHM. The input file name will be the name used under *Pre-Process_FEHMN*, or it can be changed with the *Name_Input_File* button under *Process_FEHMN*.

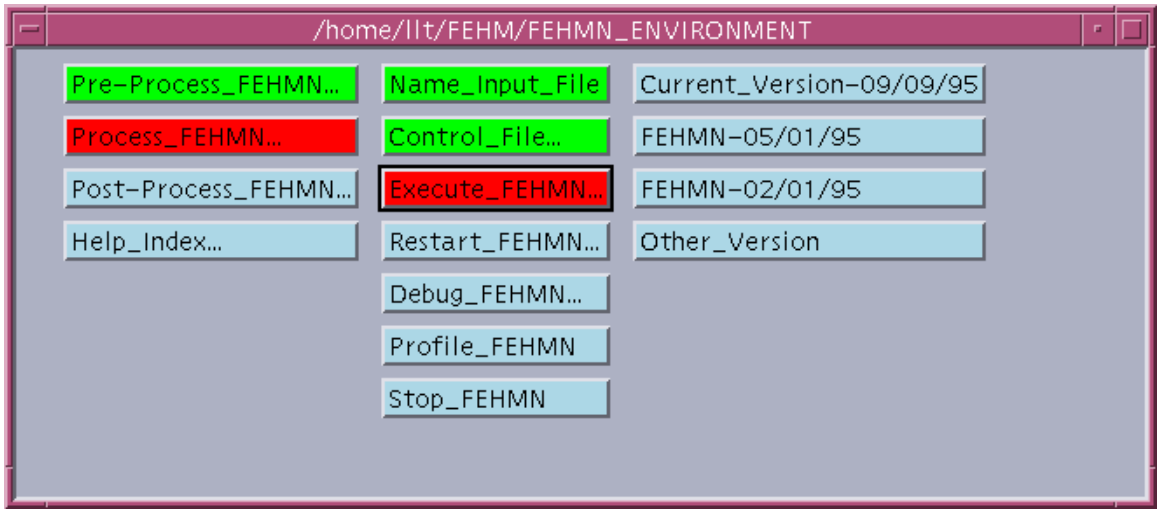


- Control Files

  The *Control_File* button allows users to create their own control file or have the Browser create the appropriate control file. The default control file name is *fehmn.files*. Another name can be entered with the *Name_Control_File* button. The *Edit_Control_File* button allows the user to directly edit the control file. The *Load_Default_Control_File* button will create a control file based on the name of the current input file. The *Read_GENMESH_grid* and *Read_Rectangular_grid* buttons modify (or create, if none exists) the control file to read the appropriate grid input file. The *Write_to_Output* button modifies (or creates, if none exists) the control file to write the output to a file. The *Write_to_Terminal* button modifies (or creates if none exists) the control file to write the output to the terminal. If you do not want FEHM to automatically run with a control file, fehmn.files can be deleted with the *Remove_Control_File* button.

- Executing, Restarting, Stopping FEHM

Options for the most current version of FEHM and older, frozen versions are available under the *Execute_FEHMN* button. Simply click on the version needed. If the control file *fehmn.files* is in your local space, the program will automatically begin. If another control file name is used, an interactive mode will start in an xterm window and the control file name can be entered at the first prompt. If no control file is used, the user will be prompted to input file names.



Options for the most current version of FEHM and older, frozen versions are available to continue a run under the *Restart_FEHMN* button. When these are clicked on, a help page will pop up that describes how the macro **time** needs to be adjusted for a restart run. The control file will be changed for a restart run, or if there is no control file, one will be made. The **.fin* file will be copied to **.ini*.

Options for the most current version of FEHM and older, frozen versions are also available for a debug mode and a profile mode under the *Debug_FEHMN and Profile_FEHMN* buttons, respectfully.

FEHM can be stopped at any time by using the *Stop_FEHMN* button. This button will give you the process id and directions on how to kill the process in the xterm window provided.

8.5.1.3 Browser FEHM post-processor

The Browser Post-Processor allows the user to view FEHM output files, make time history plots, make contour plots, and display the output with AVS (an AVS license is required).

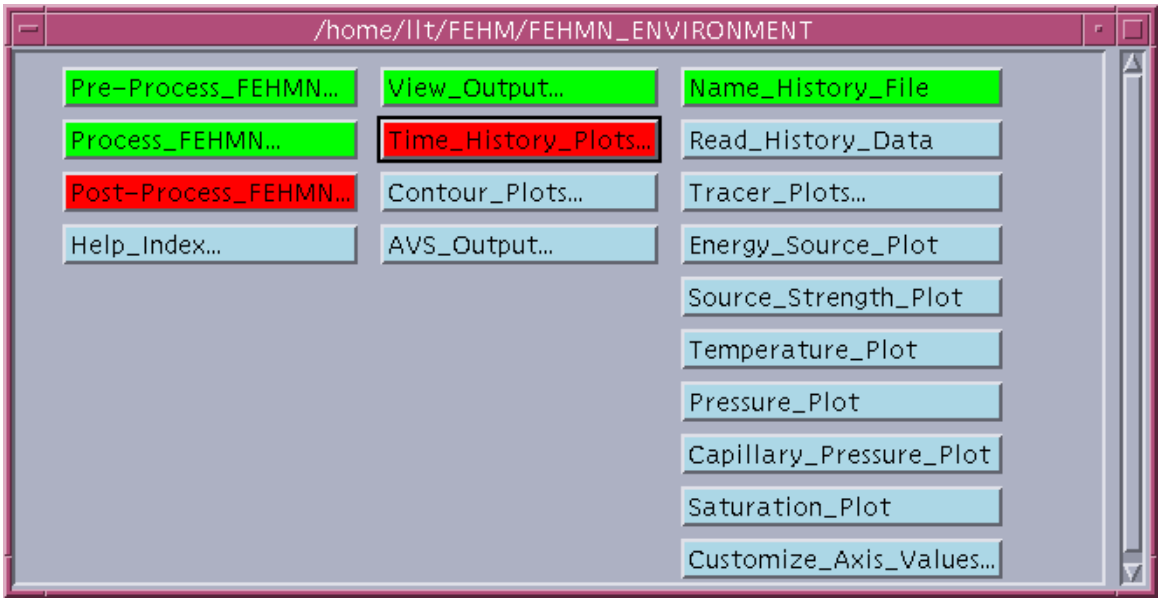
- Viewing FEHM Output Files

  The *View_Output* button will allow the user to view the FEHM output files. The name used for the * (wildcard character) will be the prefix of the last input file name used, or the name entered in the *Name_Output_File* button.

DRAFT 3/97

- Time History Plots

  The *Time_History_Plots* button allows the user to make time



history plots of the energy source, source strength, temperature, pressure, capillary pressure, and saturation from the FEHM **.his* and time history Tracer plots of the 10 species concentrations from FEHM **.trc* output files. **.his* is specified on the seventh line of the control file, and **.trc* on the eighth line (see Section 6.2.1 for more information on how to set up your control file to create these output files). A line for each node specified in the input is plotted as time versus the value. The plots can be zoomed in, the window can be resized, and a log scale can be used for the x- or y-axis.

The title on the first line of the input deck is used for the title of the plot. The second title line displays the version of FEHM and the date run. The x and y axes are automatically scaled. A legend is automatically displayed showing which line corresponds to which node.

The default time history file is *fehmn.his*. To change this, the user should click on the button *Name_History_File*. The data will automatically be converted to xrt plotting format when a new history file is named. To read in new data with the same file name (or if using the default), select the *Read_History_Data* button. This action will convert the new data to the xrt plotting format.

Once the data are converted, plots can automatically be brought up by selecting the button for the plot desired. The *Tracer_Plots* button will bring up another column listing the 10 species. Only the species in the **.trc* file will be available for plotting. A message will be given if a species is selected that was not in the **.trc* file.

The values for the axis can be specified. Select the *Customize_Axis_Values* button, the *New_XY_Values* button, then the axis you want to specify. You can get the minimum and maximum values or return to having it automatically specified with the *Default_XY_Values* button. After setting the axis values, select the *Save_Axis_Values* button to save them.

Following are the controls available to manipulate the time history plots:

Zooming: Press control and hold down the left mouse button. Move the mouse to draw a box around the area to zoom into.

Scaling: Press control and hold down the middle mouse button. Move the mouse down to increase the graph's size. Move the mouse up to decrease the graph's size.

Reset: To reset the window, type r.

Resizing Window: The window size can be changed by clicking on a corner of the window and dragging the mouse to the desired size.

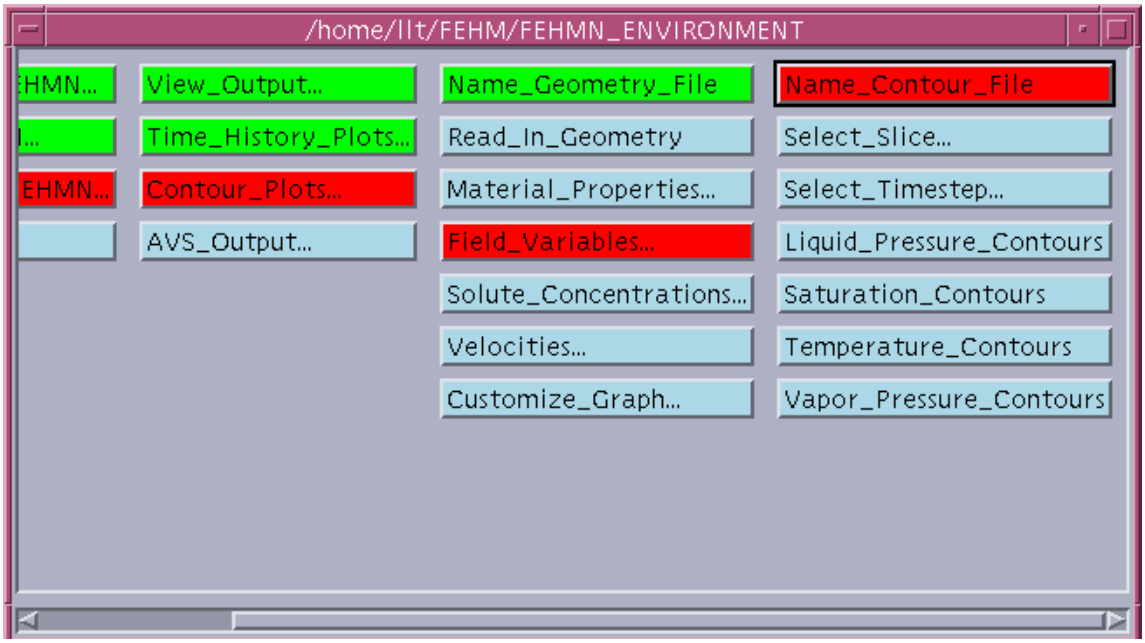
Log Scale: The x and y axes can each be changed to a log scale by clicking on the toggle button on the top row.

Printing: Click on the *Print* button on the top row. The default is set for the printer named *graphics*. This default can be changed by editing the name. A postscript file or a XWD file can also be printed. Other options can be changed under the *Properties* button along the bottom.

Exiting: Click on the *Exit* button on the top row.

- Contour Plots

The *Contour_Plots* button allows the user to make plots of x, y, or z slices of material properties, temperature, saturation, pressure, velocities, and solute concentrations for 2-D quad and 3-D hex grids. The plots can be rotated freely in the x-, y-, or z-direction or from any end point. The plots can be zoomed, scaled, translated, and the window can be resized.



There are options for drawing the mesh, drawing shaded surfaces, drawing contour lines, and drawing zones.

The title on the first line of the input deck is used for the title of the plot. The version of the FEHM executable and date run is the second line of the title. A title at the bottom of the plot includes the type of plot, the slice taken, and which time step, if applicable. The x-, y-, and z-axis and the color bar are automatically scaled, unless specific values are specified. A legend is automatically displayed showing the numeric value of the colors.

The contour plots use data from FEHM AVS output files created by specifying the **cont** macro with **avs** option in the FEHM input file (see Section 6.2.7). Slices of material-property contour plots can be made by specifying **material** after **avs**. The contour plots include permeability in the x-, y-, or z-direction; thermal conductivity in the x-, y-, or z- direction; porosity; rock specific heat; capillary pressure; and models for relative permeability and capillary pressure.

Time-dependent slices of field-variables plots can be made for liquid pressure, vapor pressure, saturation, and temperature. A combination of **pressure** and **liquid** or **vapor**, **saturation**, **temperature** after **avs** in the **cont** macro is needed.

Time-dependent slices of solute concentrations can be made by specifying **concentration** after **avs** in the **cont** macro. Buttons for the 10 species are provided.

Time-dependent slice of velocities can be made by specifying **velocity** and **liquid** or **vapor** after **avs** in the **cont** macro.

The speed (the square root of the sum of the squares of the x, y, and z velocities) for liquid and vapor can be displayed.

The default AVS geometry file is *fehmn.10001_geo*. To change this, the user should click on the button *Name_Geometry_File*. The geometry will automatically be converted to xrt plotting format when a new geometry file is named. To read in new data with the same geometry file name (or if using the default), select *Read_In_Geometry*. This action will convert the new data to the xrt plotting format. The same geometry can be used with multiple contour files.

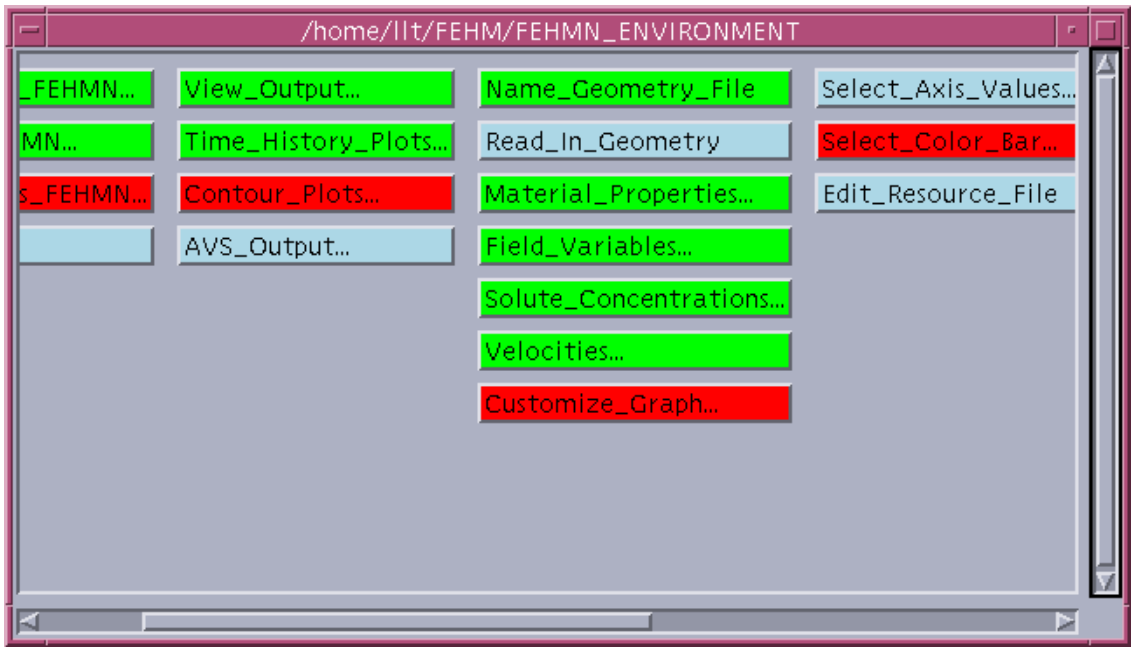
The default AVS contour files are *fehmn.1000**. To change this, the user should click on the button *Name_Contour_File*. The contour data will be read from the file selected.

Once the geometry is converted and a contour file named, plots can automatically be brought up. For material-property plots with 3-D grids, the user must select the type of slice and the value for the slice. If not selected, the default will be a xy-slice with a z-value of 1.0. The geometry values (from the file **.10001_geo*) can be displayed by clicking on the button *Display_Geometry_File*. To display the plot, the user should click on the plot type. For 3-D grids, the slice type and value will remain the same for additional plot types until changed.

For field-variable, solute-concentrations, and velocity plots with 3-D grids, the user must select the type of slice and the value for the slice. If not selected, the default will be a xy slice with a z-value of 1.0. For 2-D and 3-D grids, the user must also select the time step (the default is the second time step). The applicable time steps (from the file **.10001_avs_log*) can be viewed by clicking on the button *Display_Time_Steps*. The user can enter the time step desired with the *Input_Time_Step* button or the value of the day desired with the *Input_Number_ Days* button. To display the plot, the user should click on the plot type. The slice type, value, and time step will remain the same for additional plot types, until changed.

The values for the axis can be specified. Select the *Customize_Graph* button, then the *Select_Axis_Values* button and the axis you want to specify. You can get the minimum and maximum values or return to having it automatically specified.

The number of colors and the colors can be specified. Select the *Select_Color_Bar* button. The colors for the color bar can be selected from a range of colors with the *Select_Colors* buttons, and the number of color divisions can be selected with the *Change_Number_Color_ Divisions* button.



Other available color buttons are *Use_Standard_Color_Bar*, *Use_Random_Colors*, *Use_Gray_Scale*, *Use_No_Color* (black & white). To select any color available on the system, use the *Display_Available_Colors* button; to return to having automatically generated color bars, use the *Use_Auto_Color_Bar* button. The files containing the list of colors and the number of color divisions can also be directly edited with the *Edit_Color_List* button and the *Edit_Color_Divisions*. Colors are appended to the color list when the *Select_Color* buttons are selected. To pick up new colors, click on the *Edit_Color_List* button and delete the list contents of this file first.

With the *Edit_Resource_File* button, the graph and axis titles can be modified. The rotation of the graph can also be modified.

Following are the controls available to manipulate the contour plots:

Zooming: Press control and hold down the left mouse button. Move the mouse to draw a box around the area to zoom into.

Scaling: Press control and hold down the middle mouse button. Move the mouse down to increase the graph's size. Move the mouse up to decrease the graph's size.

Translation: Press shift and hold down the middle mouse button. Move mouse to shift the graph.

Reset: To reset the window (for zooming, scaling, and translation only), type r.

Rotating: Hold down the middle mouse button and either:

- move mouse counter-clockwise to rotate view clockwise or

- press x, y, z, or e to select an axis and then move mouse perpendicular to axis.

Resizing Window: The window size can be changed by clicking on a corner of the window and dragging the mouse to the desired size.

Printing: Click on the *Print* button, on the top row. The graph can be printed to any printer or a file. It can create postscript, XWD, or CGM files. Other options can be changed under the *Properties* button along the bottom.

Exiting: Click on the *Exit* button on the top row.

Various displays can be made using combinations of the four Toggles at the top of the display:

DrawMesh: When highlighted, displays the x-y grid projected onto the 3-D surface in a 3-D view with a z-axis. The graph honors rotation and perspective control.

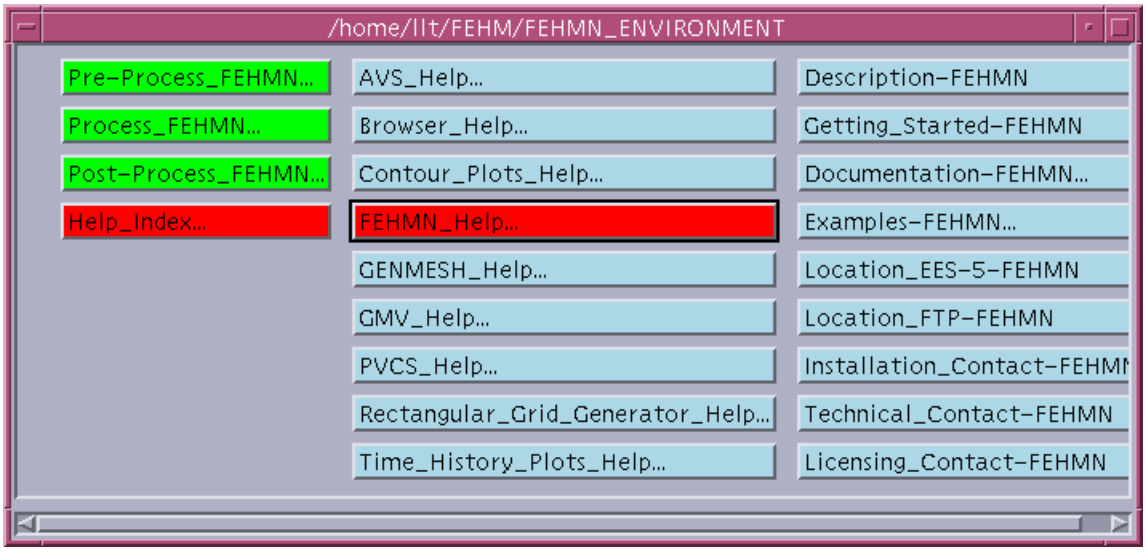
DrawShaded: When highlighted, displays the data as a flat shaded surface in a 3-D view with a z-axis. The graph honors rotation and perspective control.

DrawContours: When highlighted, examines the distribution of the data and draws contour lines demarcating each of the distribution levels.

DrawZones: When highlighted, examines the distribution of the data and fills each level with a solid color.

• Display AVS Output

The *Run_AVS* button will execute the commercial program AVS (an AVS license is required). The name of the FEHM AVS output file is entered within AVS.



8.5.1.4 Browser help index

The *Help_Index* button allows the user to view the FEHM documentation and documentation for other codes used within the Browser.

8.6 Installation Instructions

On the EES-5 computer network, no installation is required to run FEHM. Table VI lists the location and executable name for each platform.

Table VI. FEHM executable locations

Platform	Path	Executable name
Solaris	/pvcs.config/fehmn/fehmn_96_05_07/objects_sol	xfehmn
IBM	/pvcs.config/fehmn/fehmn_96_05_07/objects_ibm	xfehmn
HP	/pvcs.config/fehmn/fehmn_96_05_07/objects_hp	xfehmn
SGI	/pvcs.config/fehmn/fehmn_96_05_07/objects_sgi	xfehmn

FEHM object code is provided for users modifying a few routines to link against for installation. FEHM source code is provided for users needing the entire source and for remote users.

Remote users can obtain FEHM executable through a passworded ftp account after a software license agreement form has been completed.

8.6.1 Installation of FEHM using objects

Users that need to modify a few routines but do not need the entire source should only have the source they are changing in their local space. On the EES-5 Sun network, the following makefile should be used:

/pvcs.config/fehmn/ref/makefile

-OR-

/pvcs.config/fehmn/ref/makefile-g (for debug)

and FEHM is installed by typing:

make

-OR-

make -f makefile-g (for debug)

On the EES-5 IBM (magma), the following makefile should be used:

/pvcs.config/fehmn/ref/makefile_ibm

-OR-

/pvcs.config/fehmn/ref/makefile_ibm-g

and FEHM is installed by typing:

make -f makefile_ibm

-OR-

make -f makefile_ibm-g (for debug)

All makefiles create an executable called:

xfehmn

8.6.2 Complete installation of FEHM

Users that need the entire source and remote users can automatically setup the FEHM directory structure. On the EES-5 Sun network, get the following tar files:

/pvcs.config/fehmn/fehmn_current/src_files.tar

/pvcs.config/fehmn/fehmn_current/objects_files.tar

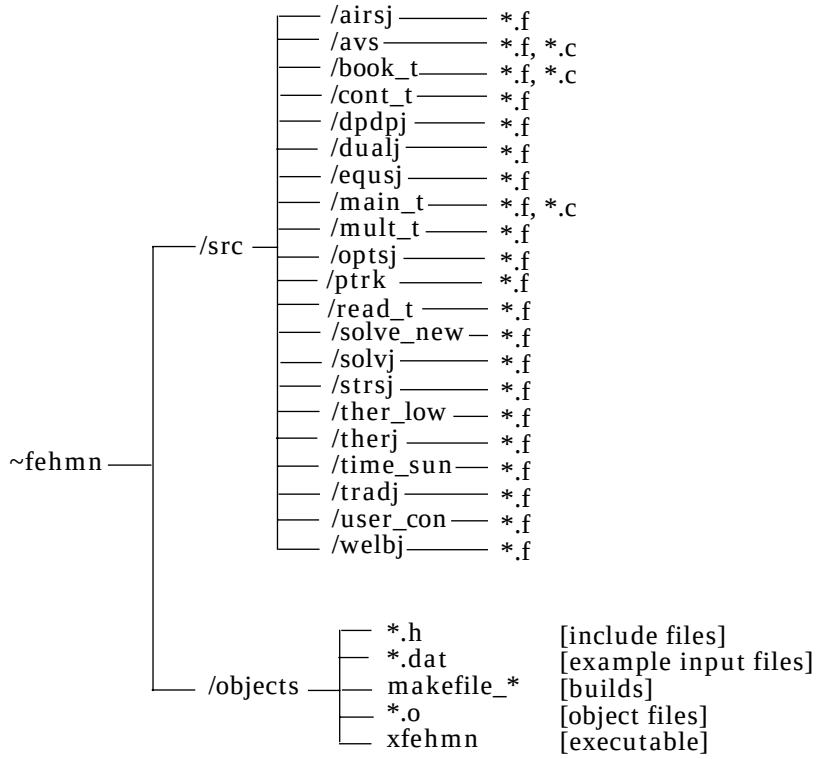
and then type:

tar xvf src_files.tar

tar xvf objects_files.tar

DRAFT 3/97

The FEHM directory structure should look like the following:



Makefiles are included in these tar files and will be placed in your objects directory. There are makefiles for various platforms with options for debug, pvcs configuration builder commands, and purify. The format is:

makefile_*machine type*{_pvcs or nopvcs_}[_purify_][-g]

Machine_type can be cray, hp, ibm_long, sgi, or sun. The options inside {} are only available with the sun makefiles and are required. The items inside [] are optional. The purify option is to link in purify and is only available with the sun makefiles. The -g option is to link in debug and is available on all machines. Following are some examples:

makefile_ibm_long

makefile_hp-g

makefile_sun_nopvcs

makefile_sun_pvcs-g

makefile_sun_pvcs_purify-g

To compile and link FEHM, select the appropriate makefile and type the following:

cd ~fehmn/objects

make -f makefile_sun_nopvcs -OR- make -f makefile_hp

All makefiles create an executable called:

xfehmn

It should be noted that FEHM uses the GZSOLVE application reuse components, solve_new and slvesu (Zyvoloski and Robinson 1995). Please refer to the SOLVE_NEW UG component of the document just cited for information on installation of the solver routines.

8.6.3 Installation verification and validation

A series of test scripts have been developed to automate the verification and validation procedure for FEHM. They are described in more detail in the Appendix of "Software Verification and Validation for the FEHM Application" (Dash et al. 1997). Also, see this document for a discussion of the tests performed and their results.

8.6.4 FEHM licensing at remote sites

A software license agreement must be completed before receiving an FEHM executable. To obtain access, get either the Commercial_agreement.ps or Government_agreement.ps form from anonymous ftp:

ftp ees5-ftp.lanl.gov

Name (): anonymous

Password: [enter your email address]

cd pub/fehm

ftp> binary

ftp> get Commerical_agreement.ps

or

ftp> get Government_agreement.ps

The form can be printed with any postscript printer. After reading, please sign the form and return to:

Lynn Trease

Mail Stop F665

P.O. Box 1663

Los Alamos National Laboratory

Los Alamos, NM 87545

FAX: 505-665-3687